Modeling of centrifugal casting processes with complex geometries

Humphreys, N.J.; Mcbride, D.; Croft, T.; Shevchenko, D.M.; Green, Nicholas; Cross, M.

DOI:
10.1002/9781118364697.ch22

MODELLING OF CENTRIFUGAL CASTING PROCESSES WITH COMPLEX GEOMETRIES

N. J. Humphreys[2], D. McBride[1], T.N. Croft[1], D. M. Shevchenko[2], N. R. Green[2], M. Cross[1] and P. Withey[3]

[1]College of Engineering, Swansea University, Singleton Park, Swansea, Wales, UK
[2]School of Metallurgy and Materials, University of Birmingham, Birmingham, UK
[3]Rolls-Royce Plc, P.O. Box 31, Derby, DE24 8BJ. UK

Abstract

Centrifugal casting offers a route to high quality products in difficult to cast high temperature low superheat alloys and thin section moulds. Under centrifugal forces metal is forced into thin sections and can fill thicknesses of less than a millimetre. However, due to the high liquid metal velocities there is a high risk of surface turbulent flow and air entrainment within the liquid metal. The combination of interacting flow-thermal-solidification phenomena and associated defects is a challenging modelling task which the authors have previously described and validated. Capturing the metal-air interface, on what are inevitably complex three dimensional geometries, results in highly computationally expensive simulations and simulating a single cast can take weeks on a single processor. This contribution reports on modelling a complex centrifugal cast, gas entrainment, bubble transport and solidification, employing meshes of up to a million elements and investigates the scalability of the model on high performance clusters.

Introduction

The ultimate goal of any casting procedure is to produce a defect-free product. For difficult to cast alloys, such as Titanium, centrifugal casting has a number of advantage over conventional gravity methods [1]. In the casting of thin sections, such as turbine blades, the metal is forced into the thin sections under centrifugal forces. The centrifugal force can overcome the backpressure due to surface tension and the metal can fill mould sections with thicknesses substantially less than a millimetre. However, the materials used in these castings [2] tend to be highly reactive and requires low superheat. High liquid metal velocity gives rise to a high risk of free surface turbulent flow and of associated entrainment through free surface turbulence [3] and trapping of gas present in the flow [1].

The fluid dynamics of centrifugal casting is very different from gravity casting. The turbulence of the melt inside the mould is much higher is centrifugal casting. The geometries are often very complex and defects can arise as a consequence of air becoming entrained during the filling process, remaining trapped within the mould once it has filled, and from gas pore growth as dissolved gas precipitates from solution during solidification [3].

Capturing these free surface flows and associated defects, coupled with centrifugal forces, heat transfer and solidification makes this a significant computational modelling challenge. There is now a suite of shape casting processes that involve rotating moulds to utilise the resulting centrifugal forces to control the flow dynamics and reduce the defects. Simulation of liquid metal flow during centrifugal casting has been reported in [4-11]. All these papers show that the

fluid filling process plays a crucial role in determining the casting quality. As such, the, gas and liquid phase needs to be adequately resolved in order to capture bubble formation, entrainment and transport within the liquid phase and entrapment within solidifying regions. Volume-of-Fluid (VOF) techniques are commonly used to capture the emerging free surface. However, methods that work well for relatively quiescent mould filling, such as, van Leer [12] based surface capturing algorithms smear unacceptably for these more vigourous flows. Methods which are better at surface capture, such as, donor-acceptor algorithms are much more sensitive to mesh quality and so make demands on the element quality throughout the whole mesh – one poor quality element can cause the whole procedure to fail [13]. A switching algorithm has been implemented that calculates the orthogonality of a cell face and switches from donor-acceptor to van Leer interpolation scheme on highly non-orthogonal faces.

The model has been validated against a series of water experiments [14], a series of aluminium alloy castings [15] and a benchmark TiAl solidification test [16]. This contribution reports a number of key features that have been identified and which need to be incorporated into the modelling and design of a complex centrifugal cast. A large scale complex centrifugal cast is reported on, showing the capture of gas entrainment, bubble transport and solidification. Simulations were performed employing meshes of up to a million elements on high performance clusters.

Computational Model

The model is implemented within PHYSICA [17] where the numerical procedures are based upon finite volume methods on unstructured heterogeneous meshes for complex 3D geometries [18] and with scalable speed-up in parallel [19].

The metal and air is represented as a Newtonian fluid by the Navier-Stokes equations:
for fluid momentum

$$\frac{\partial(\rho \underline{u})}{\partial t} + \nabla.(\rho \underline{\underline{u}}\,\underline{u}) = \nabla.\left(\mu_{eff} \nabla \underline{u}\right) + \underline{S}_u - \nabla p \qquad (1)$$

for mass conservation

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \underline{u}) = S_m \qquad (2)$$

where $\underline{u}$ is the fluid mixture velocity, ρ is the fluid mixture density, p is pressure and μ_{eff} is the effective viscosity. These equations may well be augmented by turbulence models to calculate the effective viscosity – although, there is much debate about their efficacy in the context of many mould filling scenarios where the opportunity to develop turbulence structures within the physical timeframe is very limited.

The governing equations are solved in a non-inertial reference frame, where the co-ordinate system moves with the rotating equipment. To account for the acceleration of the fluid the centrifugal and Coriolis forces enter the momentum equations as fictitious forces. The velocity of the fluid relative to the co-ordinate system can be expressed as;

$$\underline{u}_r = \underline{u} - (\underline{\Omega} x \underline{r}) \qquad (3)$$

Where u_r is the relative velocity, Ω is the angular velocity and r is the positon vector.

Substituting (0 into (1) and re-arranging the left hand side of (1) can be written as;

$$\frac{\partial}{\partial t}\left(\rho \underline{u}_r\right) + \nabla.\left(\rho \underline{u}_r \underline{u}_r\right) + \rho\left(2\underline{\Omega} x \underline{u}_r + \underline{\Omega} x (\underline{\Omega} x \underline{r})\right) \tag{4}$$

and equation (2) as;

$$\frac{\partial \rho}{\partial t} + \nabla.\left(\rho \underline{u}_r\right) = S_m \tag{5}$$

When the fluid is rotating at the same velocity as the rotating frame the forces should balance and the relative velocity is equal to zero. The differencing scheme employed in the numerical discretisation of the governing equations is crucial to ensuring sufficient accuracy of the flux calculations and hence the correct time period for a stationary fluid to achieve its rotational velocity. A first order scheme is not sufficient and a higher order scheme, such as, SMART [20] is required. Almost all viable simulation tools for the simulation of free surface flows in shape casting use a fixed grid approach as defined by the mould geometry. They either employ the concept of volume of fluid (VOF), originally proposed by Hirt and Nichols [21], to track the interface and to reconstruct geometrically its location, or the equation for the advection of the scalar,

$$\frac{\partial \phi}{\partial t} + \underline{u}_r.\nabla(\phi) = S_\phi \tag{6}$$

is solved numerically and then the interface is captured as a discontinuity in the solution field via an appropriate non-diffusive scheme (e.g. van Leer [12] in the SEA algorithm). The VOF tracking method leads to a sharp interface, but it is not guaranteed to conserve mass. In contrast, the surface capturing approach, although conservative, is computationally more demanding and it suffers from numerical diffusion. There are a host of workers who have essentially employed the VOF concept within algorithms using all kinds of combinations of finite element and finite volume methods. Since both methods require an explicit time stepping for accuracy, a stable solution is ensured only by using a small enough time step to satisfy the smallest CFL number in the mesh. There are now many useful tricks for ensuring this can be done efficiently. The method employed in this paper is essentially the FV-UM scheme of Pericleous [22], which uses as part of the SEA TVD capturing scheme, the GALA scheme developed by Spalding [23] combined with a swithing algorithm that employs van Leer [12] interpolation scheme on highly non-orthogonal faces to maintain stability and donor acceptor [24] to ensure the interface remains sharp as it convects through a fixed mesh. The inclusion of surface tension effects, on the interface between materials, is accounted for by a source term in the momentum equations which depends on the value of the surface tension, the curvature of the interface and any effects caused by the interaction of the free surface and a boundary [25].

The heat transfer and solidification are described by:

$$\frac{\partial}{\partial t}(\rho\, cT) + \nabla.(\rho\ c\ \underline{u}\ T) = \nabla.(k \nabla T) + S_h \tag{7}$$

where c is the mixture specific heat capacitance and k the conductivity.

The source term S_h can represent viscous dissipation, heat due to fluid bulk motion, boundary heat transfer and the latent heat release during phase change. If only change of phase is considered, the source into the heat equation due to the solidification process is equal to:

$$S_h = -\frac{\partial(\rho f_L L)}{\partial t} - \nabla.(\rho \underline{u} f_L L) \tag{8}$$

where L is the latent heat of solidification and f_L is the liquid fraction of the metal component of the fluid. The liquid fraction is typically a function of the metal temperature, where the liquidus temperature is the temperature at which the metal is fully liquid and the solidus temperature is the temperature at which the metal has totally solidified. A Darcy like flow retardation is applied to the fluid momentum once solidification begins.

Model Validation and Key Features

Previously published work has reported on model validation against high quality experiments, using transparent moulds with the resulting fluid flow recorded using high-speed video image capture [14-15]. The validation of heat transfer and solidification, employing Darcy type flow retardation and virtual moulds, has achieved excellent agreement with experimental results [16]. The model has been shown to accurately predict the overall flow dynamics, air entrainment and bubble transport, thus allowing the key features to be captured. Previous work has shown that capturing the pour dynamics in the model inlet boundary condition is critical in order to accurately predict the flow dynamics. Newly implemented techniques for representing the inlet flow conditions have been reported in [16]. Various transfer methods have been investigated, from central inlet pours into a cylinder to projecting the metal onto the rotating wall [14].
As slow tranquil fills are not feasible for highly reactive, low superheat metals, the mould design is crucial in obtaining a clean fill in the critical region, and complex three dimensional geometries are required to minimise turbulent bubble entrainment, and heat loss. Indirect and back filling systems have also shown to give a much cleaner fill.

Simulations

Capturing the flow dynamics and bubble transport through the mold requires very fine meshes. To resolve the transport of small air bubbles through the system can only be achieved if the element size is smaller than the bubble diameter. Figure 1, shows the air bubble capture achieved employing meshes containing elements of less than 1mm. Figure 1a), illustrates the exceptional bubble resolution achieved in a good quality mesh and Figure 1b) shows the bubble resolution in a complex runner and blade system containing meshes with poorer quality elements.

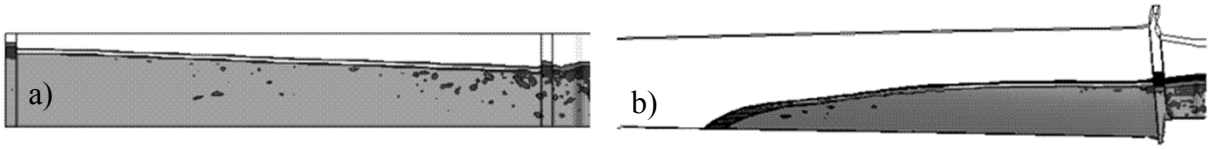

Figure 1: Bubble capture in blades a) orthogonal mesh b) complex mesh

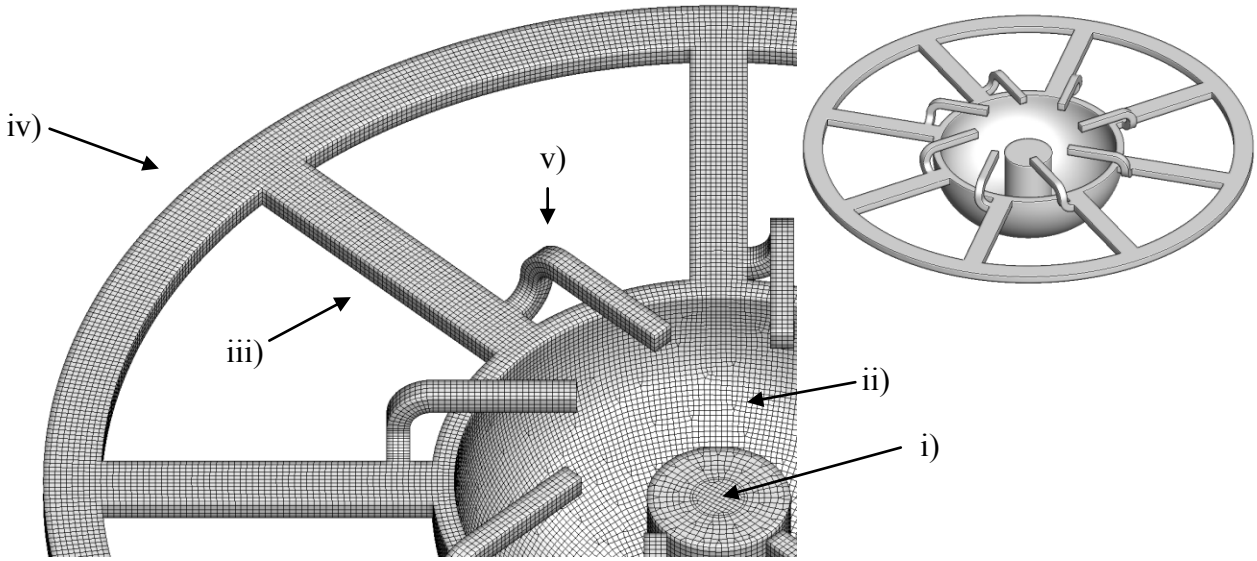

Figure 2: Test geometry mesh

Meshes containing poorer quality elements will cause some smearing of the interface, as the solution algorithm switches from the donor acceptor to van leer interpolation. This switching mechanism is performed locally on highly non-orthogonal element faces to ensure that any interface smearing is contained locally and that the interface is recovered as the fluid moves through the fixed mesh to better quality elements. Of course if the mesh contains too many poor quality elements the majority of the faces will switch to van leer interpolation resulting in significant smearing of the interface. Hence, while the method still requires high quality meshes, small sections of bad quality elements, especially if located in non-critical locations, will not degrade the solution.

Figure 2 shows a simple test geometry, designed to incorporate a number of geometry features, partially simple, but increasingly difficult to mesh in a manner that will run in a stable condition with minimal smearing. The metal is poured centrally down a sprue (Fig. 2i), then spreads across the bowl shaped surface (2ii). Both incorporate hexahedral and pentahedral element types, being swept uniformly down the sprue, before being fitted to the three dimensional bowl. The metal travels down a runner system (2iii) before collecting in the outer ring (2iv). These sections are modeled only with hexahedral elements, along the straight sections, and fitted to the ring. Finally, vents are required to expel excess gas as the mould is filled (Fig 2v)

As can be seen in Figure 3, the central pour contains small bubbles of air, consistent with previously reported inlet conditions that accurately describe the turbulent pour condition. Figure 4 shows the metal travelling down the wall delivery runners, opposite to the rotational direction, under centrifugal and Coriolis forces to fill the outer runner. As can be seen in the plots the model captures the pockets of air being expelled back up the delivery runner to leave a clean metal fill in the outer runner.

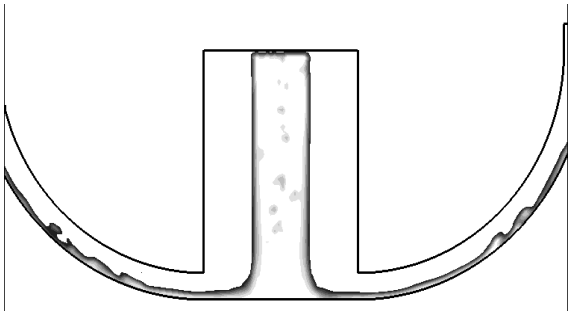

Figure 3: Bubble capture in central pour

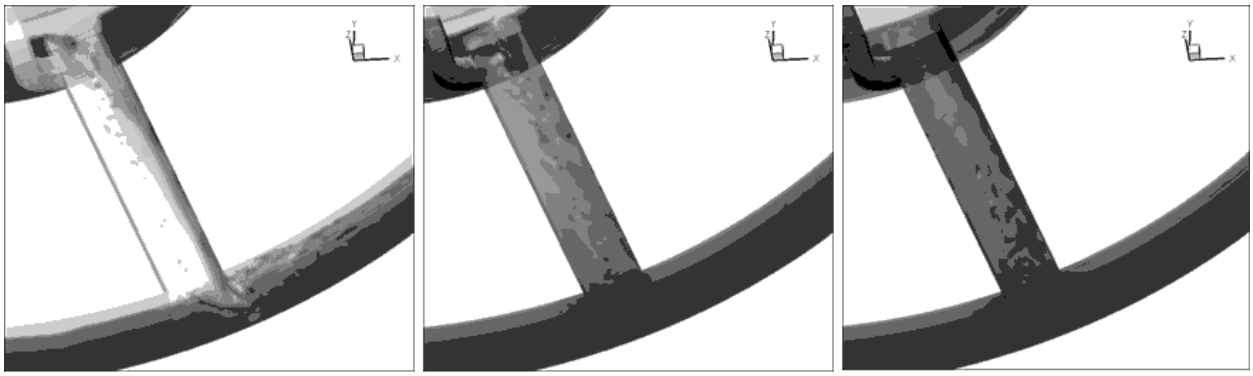

Figure 4: Plots showing metal fill and entrained air over time

Computational Performance

Employing meshes of more than a million elements and the small time steps required to satisfy the smallest CFL number in the mesh make these simulations computationally very intensive. A mesh consisting of a million elements requires 147mb of memory and would take approximately one week to run a simulation on a single processor. Techniques can be employed to speed up the simulation, such as split fluxes, where the flow field is solved implicitly employing a larger time step size and the free surface interface is advanced explicitly employing a smaller time step. However, these techniques can introduce errors into the simulation when the flow dynamics are rapidly changing.

A series of simulations were performed on Intel x5560 2.93 Mhz Nehalem processor with 10 q Myrinet switch configuration and dual quad core with 8G memory per node. The parallel code employs a JOSTLE graph partitioning and dynamic load balancing tool. This method has been specifically developed for multi-physics simulations, such as solidification melting problems, where the computational workload per mesh element is not constant [26]. Figure 5 shows the speed up achieved on multiple processors. As can be seen from the graph, ideal speed up is achieved when running on up to 6 processors, before the inter-processor latency degrades the efficiency. On 16 processors the speed up is only 20% less than ideal, however, at 32, nearly half of the ideal processing power is lost, and at 48, almost 58%.

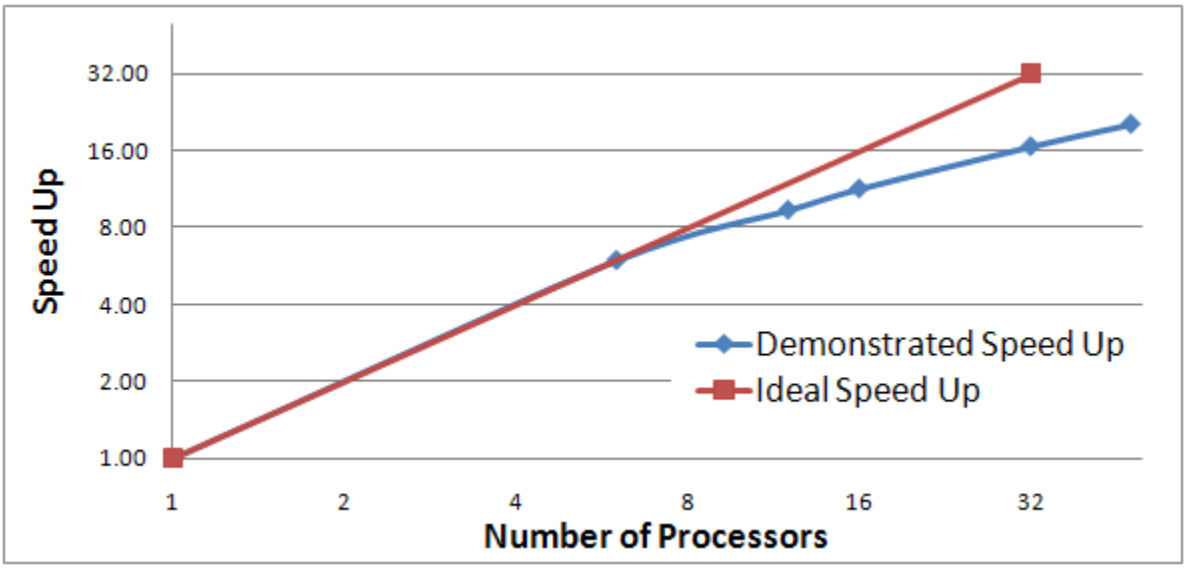

Figure 5: Speed up employing multiple processors

Conclusion

Although reliable CFD methods for the simulation of free surface flows have been established for well over 15 years on unstructured mesh codes, their application to very complex problems, especially involving centrifugal forces brings significant challenges. To effectively address such problems requires a combination of careful enhancements to the standard methods, including higher order approximations for diffusion fluxes and stable non-diffusive interface capturing methods. However, even with such enhancements there is no escape from the need for high quality meshes where all the elements are of an acceptable quality, and this has only recently been available from commercial tools. Finally, the simulation times on a single processor are significant and really mandate the exploitation of high performance clusters if reasonable times are to be achieved and enable the simulation route to be practical in the design for manufacture cycle.

References

[1] R.A. Harding, M. Wickins, and Y.G. Li, "Progress towards the production of high quality γ-TiAl castings," Structural Intermetallics 2001. Proceedings of the Third International Symposium on Structural Intermetallics, 23-27 Sept. 2001, 181-189.

[2] Wu X., "Review of alloy and process development of TiAl alloys", *Intermetallics*, 14 (2006), 1114-1122

[3] Campbell J., *Castings* (Oxford, UK: Butterworth-Heinemann, 2003), 17-69

[4] W. Shiping et al., "Numerical simulation and experimental investigation of two filling methods in vertical centrifugal casting," *Transaction of Nonferrous Metals Society of China*, 16 (2006), 1035-1040.

[5] W. Shiping et al., "Numerical simulation of microstructure evolution of Ti-4Al-4V alloy in vertical centrifugal casting," *Materials Science and Engineering*, 426 (2006) 240-249

[6] W. Shiping et al., "Numerical simulation of off-centred porosity formation of TiAl-based alloy exhaust valve during vertical centrifugal casting", *Modelling and Simulation in Materials Science and Engineering*, 11 (2003), 599-608.

[7] S. Wen-bin, "Filling and solidification of TiAl melt in centrifugal field", *Transaction of Nonferrous Metals Society of China*, 16 (2006), 719-722.

[8] S.R. Chang, J.M. Kim, and C.P. Hong, "NumericalNumerical simulation of microstructure evolution of Al alloy in centrifugal casting," *ISIJ International* 41 (2001), 738-747.

[9] L. Changyun et al., "Hydraulic modelling of mould filling behaviour during vertical centrifugal casting," *International Journal of Cast Metals Research*, 19 (2006), 237-240.

[10] P.X. Fu et al., "Centrifugal casting of TiAl exhaust valves," *Intermetallics*, 16 (2008) 130-138.

[11] Si-Young Sung, Young-Jig Kim, "Modeling of titanium aluminides turbo-charger casting," *Intermetallics*, 15 (2007) 468-474

[12] B van Leer, "Towards the ultimate conservation difference scheme. IV a new approach to numerical convection", *J Comp Phys*, Vol. **23**, pp. 276-299 (1979)

[13] S M Bounds, G Moran, K A Periclous, M Cross and T N Croft, et al., "A Computational Model for Defect Prediction in Shape Castings Based on the Interaction of Free Surface Flow", Heat Transfer, and Solidification Phenomena, *Metallurgical and Materials Transactions B*, Vol. **31B,** pp.515-527 (2000)

[14] D.McBride, N.J.Humphreys, T.N.Croft, N.R.Green and M.Cross, "Complex Free Surface Flows for Mould Filling using Centrifugal Casting", *Modeling of Casting, Welding and Advanced Solidification Processes – X11 (eds. S L Cockcroft and D M Maijer), pub TMS,* pp. 77-84 (2009).

[15] D.M.Shevchenko, D.McBride, N.J.Humphreys, T.N.Croft, P.Withey, N.R.Green and M.Cross, "Centrifugal Casting of Complex Geometries: Computational Modelling and Validation Experiments", *Modeling of Casting, Welding and Advanced Solidification Processes – X11 (eds. S L Cockcroft and D M Maijer), pub TMS,* pp. 459-466 (2009).

[16] N.J.Humphreys, D.McBride,D.M.Shevchenko,T.N.Croft,P.withey,N.R.Green and M.Cross, 'Modelling Centrifugal Casting: The Challenges and Validation', Int. Sym. On Liquid Metal Processing and Casting, Nancy, France, (2011).

[17] PHYSICA, see http://physica.co.uk

[18] K Pericleous, G L Moran, S M, Bounds, P, Chow, and M Cross, "Three Dimensional Free Surface Modelling in an Unstructured Mesh Environment for Metal Processing Applications". *Applied Math Modelling*, Vol. **22**, pp.895-906, (1998)

[19] K McManus, A J Williams, M Cross, T N Croft, and C Walshaw, "Assessing the parallel performance of multi-physics tools for modelling of solidification and melting processes", *Intl Jnl High Perf Comp Applns*, **19**, 1-27 (2005)

[20] P H Gaskell and A K C Lau, "Curvature-compensated convective transport: SMART, A new boundedness-preserving transport algorithm", *IJNME*, **8**, pp. 617-641 (1988)

[21] C W Hirt and B D Nichols, "Volume of fluid (VOF) method for the dynamics of free boundaries", *J. Comp. Phys*. **39**, pp 201-225 (1981).

[22] K Pericleous, "Three-dimensional free surface modelling in an unstructured mesh environment for metal processing applications", *Appl Math Modelling*, **22**, pp 895-906 (1998)

[23] D B Spalding, "Mathematical modelling of fluid mechanics, heat transfer and chemical reaction processes", *Report HTS – 80-1*, Mech Engg Dept, Imperial College London (1980)

[24] J B Ramshaw and J A Trapp, "A numerical technique for low speed homogeneous two phase flow with a sharp interface*, Journal of Computational Physics*, **21,** pp 438-453 (1976)

[25] D Wheeler, C Bailey, M Cross, "Numerical Modelling and Validation of Marangoni and Surface Tension Phenomena Using the Finite Volume Method", *International Journal for Numerical Methods in Fluids*, **32**, pp. 1021-1047 (2000)

[26] K McManus, A Williams, M Cross, N Croft, C Walshaw, "Assessing The Scalability of Multiphysics Tools for Modeling Solidification and Melting Processes on Parallel Clusters", *International Journal of High Performance Computing Applications*, **19**, 1, pp 1-27 (2005).